2001

**Transonic flutter computations for the NLR 7301 supercritical airfoil**

Weber, Stefan

Éditions scientifiques et médicales Elsevier SAS.


http://hdl.handle.net/10945/50273
A numerical investigation of the transonic steady-state aerodynamics and of the two-degree-of-freedom bending/torsion flutter characteristics of the NLR 7301 section is carried out using a time-domain method. An unsteady, two-dimensional, compressible, thin-layer Navier–Stokes flow-solver is coupled with a two-degree-of-freedom structural model. Fully turbulent flows are computed with algebraic or one-equation turbulence models. Furthermore, natural transition is modeled with a transition model. Computations of the steady transonic aerodynamic characteristics show good agreement with Schewe’s experiment after a simplified accounting for wind-tunnel interference effects is used. The aeroelastic computations predict limit-cycle flutter in agreement with the experiment. The computed flutter frequency agrees closely with the experiment but the computed flutter amplitudes are an order of magnitude larger than the measured ones. This discrepancy is likely due to the omission of the full wind-tunnel interference effects in the computations.

Received 14 August 2000; revised and accepted 2 February 2001

Abstract

A numerical investigation of the transonic steady-state aerodynamics and of the two-degree-of-freedom bending/torsion flutter characteristics of the NLR 7301 section is carried out using a time-domain method. An unsteady, two-dimensional, compressible, thin-layer Navier–Stokes flow-solver is coupled with a two-degree-of-freedom structural model. Fully turbulent flows are computed with algebraic or one-equation turbulence models. Furthermore, natural transition is modeled with a transition model. Computations of the steady transonic aerodynamic characteristics show good agreement with Schewe’s experiment after a simplified accounting for wind-tunnel interference effects is used. The aeroelastic computations predict limit-cycle flutter in agreement with the experiment. The computed flutter frequency agrees closely with the experiment but the computed flutter amplitudes are an order of magnitude larger than the measured ones. This discrepancy is likely due to the omission of the full wind-tunnel interference effects in the computations.

Unsteady transonic flows / Flutter / Transition-turbulence

Zusammenfassung


Durch Berücksichtigung des Einflusses der Windkanalwände auf die Strömung in Form einer korrigierten Anströmzahl und eines korrigierten Anströmwinkels konnte für die stationäre Stromung eine gute Übereinstimmung zwischen numerischer Simulation und den Meßwerten von Schewe erreicht werden. Die instationären aeroelastischen Rechnungen ergaben in Übereinstimmung mit dem Experiment Limit-Cycle-Flattern. Während die berechneten Frequenzen annähernd den experimentell ermittelten entsprachen, waren die numerisch vorhergesagten Amplituden um eine Großenordnung höher als im Experiment. Diese Abweichung ist darauf zurückzuführen, dass für die instationären Rechnungen die Windkanalwandeinfluße nicht berücksichtigt wurden. © 2001 Éditions scientifiques et médicales Elsevier SAS

Instationäre transsonische Strömung / Flattern / Transition-Turbulenz

Correspondence and reprints.
E-mail address: ekaterin@iacm.forth.gr (J.A. Ekaterinaris).
1. Introduction

The greater proclivity of typical aircraft wings to flutter at transonic flight speeds is a well recognized phenomenon. Transonic flutter may also occur on propeller and helicopter blades, and in high performance compressor and turbine stages. There is therefore a great need to develop reliable transonic flutter prediction methods. These methods must include a mathematical model which describes the physical system closely and uses efficient, robust, and sufficiently accurate numerical techniques for the solution of the governing equations. The models must be capable of predicting the strong nonlinear, viscous flow phenomena which are encountered at transonic speeds. In turbomachinery applications, one encounters the additional complication that the blade chord Reynolds number is often quite low. Therefore, improved modeling of the transitional flow effects becomes quite important. In many practical applications it is of interest to predict the full three-dimensional aeroelastic behavior. In this numerical investigation, however, we focus on the two-dimensional flutter problem studied experimentally by Schewe and Deyhle [20]. The assumptions used in the mathematical model introduce some degree of simplification but make numerical simulations feasible, cost effective, and allow the study of various factors that affect physical modeling.

We perform numerical simulations for flutter caused by transonic separated flow on the NLR7301 supercritical airfoil section. The airfoil is rigid and it is considered suspended by a two-degree-of-freedom spring/mass/damper system emulating the bending and twisting of a flexible wing. Measurements for this rigid airfoil section suspended in a wind tunnel are available from the experiments of Schewe and Deyhle [20] and Knipfer et al. [14]. The tunnel walls in the experiment were approximately a chord length away from the airfoil surface, and even though wall perforation was used to reduce shock-wave reflection effects, tunnel wall interference clearly affected both the stationary and the unsteady test results. In our investigation we simplify the problem by omitting the tunnel walls and attempt to match the measured surface pressure distribution of the stationary airfoil as closely as possible by correcting both the freestream Mach number and angle of attack. This type of correction cannot be expected to account for wind-tunnel interference effects during oscillatory motion. Therefore, the flutter calculations presented in this paper cannot be expected to yield complete agreement with Schewe’s experiment, but should be regarded as a prediction method for the NLR 7301 airfoil in free flight.

Mortchelewicz [15], Morton and Beran [16] and Dimitrievic et al. [8] analyzed subsonic/transonic airfoil flutter using the Euler equations. Recent advances in computer power make it possible to use the full viscous flow equations in aeroelastic solvers in two- and possibly three-dimensional configurations. The complicated flow physics of transonic separated flows can be captured with advanced Navier–Stokes analysis which include sophisticated transition modeling and state of the art turbulence models. Successful modeling of unsteady flow effects is accomplished by capturing the inviscid nonlinear flow features, the viscous effects caused by the state of the boundary-layer, the shock/boundary-layer interac-
tion including flow-separation, and the presence of separation bubbles. Among others, Van Dyken et al. [23] have shown that the boundary-layer transition has a significant effect on the onset of flow separation, even at moderately high Reynolds numbers. Therefore, predictions of transitional flow are essential to capture flow features, such as separation bubbles and shock/boundary-layer interactions.

Modeling of transitional and turbulent separated flow introduces physical uncertainties. In addition, by omitting the tunnel walls for the simulations of the experiment of Schewe and Deyhle [20], and matching the pressure distributions of the wing by correcting the freestream Mach number and angle of attack for the static case, does not imply that either instantaneous or time-averaged surface pressure distributions for the unsteady simulation will correspond. Therefore, complete agreement with the flutter test results cannot be expected. The objectives of the present numerical investigation are to demonstrate the capability of a coupled aero-structural dynamic solver to predict flutter based on first principles and to find out whether viscous flow effects and separation play major roles in flutter development.

The flow solver and the aeroelastic models have been tested and validated extensively in previous studies for a variety of flow conditions. For example, the flow solver has been tested for subsonic turbulent flow by Sanz and Platzer [19], Ekaterinaris and Menter [10], Ekaterinaris et al. [11], and for transonic flow by Ekaterinaris et al. [9]. The aeroelastic model has been implemented and tested by Jones and Platzer [13] for inviscid subsonic flow calculations. In the present study, the turbulence models proposed by Baldwin and Lomax [2] (BL), Baldwin and Barth [3] (BB), and by Spalart and Allmaras [21] (SA) are used to model turbulent flow regions. Transitional flow regions are computed with the transition model of Gostelow et al. [12] which scales the fully turbulent eddy viscosity of these turbulence models.

The wall effects of the wind tunnel experiment of Schewe and Deyhle [20] are taken into account in the unbounded computational domain by progressively correcting the free stream speed and the angle of attack. The degree of simplification of the flow physics that permits accurate capturing of shock/boundary-layer interactions for steady-state transonic flow is investigated first. Towards this end, inviscid and viscous solutions are computed, the effect of transitional flow modeling is investigated, and various turbulence models are tested. Numerical solutions for an airfoil free to oscillate in one- or two-degrees-of-freedom in transonic flow are obtained next. Computed solutions are compared with Schewe’s experimental data.

2. Aeroelastic solver

The aeroelastic equations can be solved numerically in a fully coupled fashion. However, the solution process can be decoupled by computing the aerodynamic coefficients first. Next, the aerodynamic coefficients obtained from the aerodynamic analysis can be treated as constants for the equations governing the structural dynamics. Thus, the decoupled structural dynamics equations are linear and can be typically advanced in time with a significantly larger time step compared to the one needed for the solution of the fluid dynamic equations. As a result, different numerical methods can be used for each set of equations. The governing equations and the numerical methods used for the solution of the aerodynamics and structural dynamics are described in the following sections.

2.1. Aerodynamics

The Reynolds averaged Navier–Stokes (RANS) equations are used to compute the complex, transonic, unsteady, aerodynamic characteristics of the airfoil. Modeling of physical processes such as transition and turbulence is accomplished using state-of-the-art models. The numerical solution of the governing equations is obtained with an implicit, shock-capturing, high-resolution scheme. The essential features of the aerodynamic solver are summarized in the following section. Further details can be found in the original references.

2.1.1. Governing equations

The unsteady, compressible, two-dimensional, thin-layer Navier–Stokes equations in the strong conservation-law form and curvilinear coordinate system ($\xi$, $\zeta$) are

$$\frac{\partial}{\partial \xi} \mathbf{Q} + \frac{\partial}{\partial \zeta} \mathbf{F} + \frac{\partial}{\partial \zeta} \mathbf{G} = Re^{-1} \frac{\partial}{\partial \zeta} \mathbf{S},$$

(1)

where $\mathbf{Q} = \{\rho, \rho u, \rho w, e\}$ is the vector of conservative variables, $\mathbf{F}$ and $\mathbf{G}$ are the inviscid flux vectors, $\mathbf{F} = \frac{1}{2}(\rho U, \rho u U + \xi \rho f, \rho w U + \zeta p, (e + p)U - \xi f p)$, and $\mathbf{S}$ is the thin-layer approximation of the viscous fluxes in the $\zeta$ direction (normal to the airfoil surface). The terms $U$ and $W$ are the contravariant velocity components and the pressure is related to the other variables through the equation of state for an ideal gas $p = (\gamma - 1)(e - \rho (u^2 + w^2)/2)$. All aerodynamic quantities are nondimensionalized using $c$ as the reference length, $a_\infty$ as the reference speed, $c/a_\infty$ as the reference time, $\rho_\infty$ as the reference density, and $\rho_\infty a_\infty^2$ as the reference energy.

2.1.2. Boundary conditions

For inviscid flow solutions over stationary airfoils, the flow-tangency boundary conditions are used at the surface. The no-slip condition is applied for Navier–Stokes solutions. Density and pressure are extrapolated to the wall for both Euler and Navier–Stokes solutions.
For unsteady airfoil motions, the local motion of the surface also contributes to the pressure on the surface. Therefore, the flow-tangency and non-slip boundary conditions are modified by solving the momentum equation normal to the surface ($\zeta$ direction) to predict the pressure

$$ \partial_t p|_{wall} = -\frac{1}{\gamma} \partial_\zeta \left[ \rho \, \partial_t \left( \hat{x}|_{wall} \right) \cdot \nabla \zeta \right] + \partial_\zeta p|_{wall} \nabla \cdot \zeta. \quad (2) $$

At the farfield boundaries of the computational domain free-stream boundary conditions are imposed. The unknown variables are obtained using one-dimensional Riemann invariant extrapolation.

2.1.3. Numerical implementation

The numerical algorithm was developed and validated in [9–11]. It performs time marching with the implicit, factorized, iterative Beam and Warming [4] algorithm. The inviscid fluxes, $F$ and $G$, are evaluated using Osher’s third-order upwind-biased scheme (Osher and Chakravarthy [17] and Chakravarthy and Osher [3]). Linearization of the left-hand side is performed by evaluating the flux Jacobian matrices, $A$ and $B$, with the Steger–Warming [22] flux-vector splitting. The viscous fluxes are computed with second-order central differences. Furthermore, a standard minmod TVD scheme [17] is used to eliminate numerical oscillations at flow discontinuities, such as shocks.

Time accuracy is improved by performing Newton subiterations to convergence within each physical time step. These subiterations minimize the linearization and factorization errors and help drive the left-hand-side residuals to zero within each physical time step. Numerical experiments have shown that larger CFL numbers (i.e., a larger time step) can be used if the number of Newton iterations is increased. The optimum seems to depend on the grid density and flow conditions, but the best computational performance appears to occur with 4 to 5 subiterations on coarse grids (Euler simulations), and 2 to 3 sub-iterations on fine grids (Navier–Stokes simulations). The Navier–Stokes solver has been tested extensively in a variety of unsteady subsonic and transonic flow studies such as by Ekaterinari et al. [9].

The turbulence modeling is based either on the standard algebraic eddy viscosity model of Baldwin–Lomax [2] or the one-equation models of Baldwin–Barth [3] or Spalart–Allmaras [21]. The eddy viscosity obtained from these turbulence models is used for the computation of the fully turbulent region. The transitional flow region computed with an ‘effective eddy viscosity’ is explained in the next section.

2.1.4. Transition modeling

Transitional flow can be predicted from first principles or by using models which involve a minimum degree of simplification, such as the parabolized stability equation (PSE) method. However, in numerical simulations one is always faced with a tradeoff between modeling accuracy and computational cost. As a result, in engineering analysis the only economically viable approach is modeling of the flow in the transitional flow region by either applying a scaling of the eddy viscosity computed by a turbulence model with an intermittency function, or by modifying the damping functions included in turbulence models in order to act as an intermittency. In both cases, evaluation of the location where transition has started is required. The transition onset location can be evaluated using the $e^\theta$ method. Again for the sake of computational efficiency, the transition onset location can be obtained from existing empirical formulas which predict it as the location where the local Reynolds number exceeds a ‘critical Reynolds number’. An empirical formula for transition onset was given by Michel [7]. This criterion is based on correlation of subsonic flow measurements. Another criterion for transonic turbomachinery flows was proposed by Abu-Ghannam and Shaw [1]. This transition location prediction formula is not suitable for low free-stream turbulence levels where by-pass transition does not occur. Therefore, Rodi and Schöning [18] proposed a transition onset criterion where transition occurs only in the presence of flow separation. In our study the turbulence level is low and flow transition may occur even in the absence of flow separation. Therefore, we use Michel’s criterion for the prediction of transition onset even though it is expected that this model may be deficient for transonic flows.

The transition model of Gostelow et al. [12] was used with algebraic and one-equation turbulence models in previous studies by Sanz and Platzer [19]. This transitional-flow model was introduced in order to account for the effects of pressure gradient and free-stream turbulence level in the evaluation of the streamwise extent of the transitional flow region. This method continuously adjusts the turbulent spot growth in response to changes of the local pressure gradient.

The eddy viscosity obtained by assuming fully turbulent flow is scaled by the following intermittency function in the transitional region

$$ \gamma(x) = 1 - \exp \left[ - n \frac{\sigma}{\tan \epsilon} \left( \frac{dx}{U_o} \right) \int_{x_t}^{x_f} \tan \epsilon \, dx \right], \quad (3) $$

where the correlations for the variation of the spot propagation parameter, $\sigma$, and the spot spreading half angle, $\epsilon$, as functions of the pressure gradient parameter, $\lambda_{th}$, are:

$$ \epsilon = 4 + \frac{22.14}{0.79 + 2.72 \exp(47.63 \lambda_{th})} \quad (4) $$
\[ \sigma = 0.03 + \frac{0.37}{0.48 + 3.0 \exp(52.9 \lambda_{\theta})}. \tag{5} \]

Here \( \lambda_{\theta} = (\theta^2/v)/(dU/\text{dx}) \) with the boundary-layer momentum thickness, \( \theta \), and \( U_\text{e} \) is the outer-edge velocity. The spot generation rate, \( n \), is inferred from the dimensionless breakdown-rate parameter, \( N \), where
\[
n = \frac{vN}{\sigma \lambda_{\theta}^3}, \tag{6} \]
\[
N = 0.86 \times 10^{-3} \exp(2.134 \lambda_{\theta} \ln(q_t)) - 59.23 \lambda_{\theta} - 0.564 \ln(q_t)), \quad \text{for} \quad \lambda_{\theta} \leq 0 \tag{7} \]
and
\[
N = N(\lambda_{\theta} = 0) \times \exp(-10 \sqrt{\lambda_{\theta}}), \quad \text{for} \quad \lambda_{\theta} > 0, \tag{8} \]
and where \( q_t \) denotes the level of free-stream turbulence.

The spot-propagation rate and the spot spreading half-angle asymptotically approach a maximum value for high negative values of \( \lambda_{\theta} \), but \( n \) is allowed to increase to infinity for high negative values of \( \lambda_{\theta} \), where \( \lambda_{\theta} \) is the pressure gradient at the transition onset location, \( x_t \). The function, \( \gamma(x) \), is zero for \( x < x_t \), it increases downstream from the transition point, and reaches asymptotically the maximum value of unity, which corresponds to fully turbulent flow. An effective eddy-viscosity for the transitional region is obtained by scaling the computed turbulent eddy-viscosity by \( \gamma(x) \), i.e. \( \mu_{\text{trans}} = \gamma(x) \mu_{\text{turb.}} \).

Sanz and Platzer [19] have used the Gostelow model, originally developed for attached flow, for the prediction of laminar separation bubbles by using the spot-generation rate as a second adjustable parameter along with the location of transition onset. They investigated the influence of the spot-generation rate on the separation bubble by either limiting the breakdown-rate parameter to 1.0, which forces instantaneous transition, or by assuming the value for a zero pressure-gradient. In the present study, a breakdown-rate parameter of 1.0 was chosen, and the transition onset was either predicted by the Michel [7] criterion or by specification as an input parameter.

2.2. Structural dynamics

Structural modeling is performed by the two-degree-of-freedom aeroelastic configuration for a rigid airfoil suspended by a spring/mass/damper system of figure 1 which simulates the bending and twisting of a wing. The equations governing the motion of the two-degree-of-freedom spring/mass/damper system are:
\[
m\ddot{h} - S_a \dot{\omega} + D_h \dot{h} + m\omega_h^2 \dot{h} = L, \tag{9} \]
and
\[
-S_a \ddot{\omega} + D_\omega \dot{\omega} + I_\omega \omega_\omega^2 (\alpha - \alpha_0) = M, \tag{10} \]
where the dots denote differentiation with respect to time. These equations are nondimensionalized using the reference length \( c \), the reference velocity \( a_\infty \), the reference mass \( \rho_\infty \pi (c/2)^2 \), and the reference inertia \( \rho_\infty \pi (c/2)^2 c^2 \). Rewriting equations (9) and (10) in matrix notation where \( [M] \) is the mass, \( [D] \) the damping, \( [K] \) the spring stiffness matrix, \( [X] \) = \( \{h, \alpha - \alpha_0\}^T \) is the vector of the unknowns, and \( [F] \) is the forcing function one obtains
\[
[M][X]'' + [D][X]' + [K][X] = [F], \tag{11} \]
where the coefficient matrices are given by
\[
[M] = \begin{bmatrix} m & S_a \\ S_a & I_a \end{bmatrix},
\]
\[
[D] = \begin{bmatrix} 2\delta_h mk_h & 0 \\ 0 & 2\delta_\omega I_\omega k_\omega \end{bmatrix},
\]
\[
[K] = \begin{bmatrix} mk_h^2 & 0 \\ 0 & I_\omega k_\omega^2 \end{bmatrix},
\]
and
\[
[F] = \frac{2}{\pi} M_\infty \begin{bmatrix} -C_l \\ C_m \end{bmatrix}.
\]

**Figure 1.** Schematic of the spring/mass/damper system.
where the primes denote differentiation with respect to nondimensional time, \( \tau = t a_\infty/c \), and the other parameters (\( m \), \( I_\alpha \), etc.) are now non-dimensional. Note that \( k_\theta \) and \( k_\alpha \) appearing in the matrices \([K]\) and \([D]\) are reduced natural frequencies based on the free-stream speed of sound, as opposed to the conventional form presented in the nomenclature. However, in the interest of clarity, presented results utilize the conventional definition, based on free-stream velocity.

2.2.2. Numerical implementation

Equation (11) is a system of two coupled, second-order, ordinary differential equations. Coupling is obtained through the terms containing \( S_\alpha \) and the dependence of \( C_l \) and \( C_m \) on \( h \) and \( \alpha \). The system is nonlinear through the nonlinearity of \( C_l \) and \( C_m \). Linearization is introduced by treating \( C_l \) and \( C_m \) as constants, computed from the previous time-step of the aerodynamic solution. Simulations with a single-degree-of-freedom may be performed by setting \( S_\alpha = 0 \), and either \( m = \infty \) and \( \omega_h = 0 \), or \( I_\alpha = \infty \) and \( \omega_\alpha = 0 \) for pitching-only or plunging-only motions, respectively.

Equation (11) is advanced in time by inverting the system, yielding

\[
\{X\}'' = [M]^{-1}\{F\} - [M]^{-1}[K]\{X\} - [M]^{-1}[D]\{X\}'.
\]

(12)

Using the substitution \( \{X\}' = \{Y\} \), a system of two coupled, first-order equations is obtained

\[
\{Y\}' = [M]^{-1}\{F\} - [M]^{-1}[K]\{X\} - [M]^{-1}[D]\{Y\}'.
\]

(13)

Time integration of equation (13) is performed using the 1st-order accurate explicit Euler scheme. Higher-order methods for equation (13) were tested, such as 4th-order Runge–Kutta scheme. It was found however that the 1st-order Euler explicit scheme is sufficiently accurate because stability requirements of the Navier–Stokes flow solver impose a very small time-step. In addition use of Runge–Kutta methods makes the aeroelastic computation very intensive computationally because the aerodynamic coefficients are recomputed during each stage.

A total energy test was performed and the accuracy of the structural integration was evaluated by removing the aerodynamic influence from the problem, and releasing the airfoil with initial disturbances in \( \alpha \) and/or \( h \), essentially allowing the spring/mass system to oscillate in a vacuum, such that the total energy in the spring/mass system should remain constant in time. The kinetic and potential energy of the system was then computed as the airfoil oscillated.

The 1st-order Euler integration predicted a small oscillation in the total energy, such that the energy was periodic, with an amplitude of roughly 0.3 percent of the total energy when 1000 steps per cycle were used. The fluctuation amplitude diminished linearly with the time step-size. The Navier–Stokes solutions typically require between 1500 and 3500 steps per cycle, for this large number of steps, the energy fluctuation was deemed acceptable.

3. Results

We performed numerical simulations for the measurements obtained at experimental flow conditions of free-stream Mach number of \( M_\infty = 0.768 \) at an angle of attack \( \alpha = 1.28 \) degrees ([14,20], data for measurement no. 77). For these conditions, limit-cycle oscillations in two-degrees-of-freedom were found in the experiments of [14,20]. The experimental Reynolds number for the NLR 7301 airfoil wind-tunnel model was \( Re_c = 1.727 \times 10^6 \), based on a chord length of \( c = 0.3 \) m. In the experiment, the square shaped wind-tunnel test-section had an area of 1 m\(^2\), and the 0.3 m chord model was installed in the center. Due to this relatively large chord length, Knipfer et al. [14] corrected for steady wind-tunnel interference effects at subsonic speeds. However, no corrections for steady transonic and oscillatory interference effects were attempted. Therefore, in this paper both the free-stream Mach number, \( M_\infty \), and the angle of attack, \( \alpha_c \), were corrected until a reasonable agreement with the measured time-averaged surface pressure distribution was achieved.

All steady and unsteady viscous flow computations were carried out on a baseline, C-type, 221 \( \times \) 91 point grid. This grid was generated from the original NLR 7301 airfoil surface data taken from the University of Illinois, Champagne-Urbana, Department of Aeronautical and Astronautical Engineering, Airfoil Coordinates Database (www.uiuc.edu/ph/www/m-setig/ads/coord_database.html). A preliminary grid sensitivity investigation was performed for steady-state solutions by varying initial wall spacing and outer boundary location. As a result, a grid with an initial wall spacing of \( 8 \times 10^{-6} \), which yields \( y^+ < 1.0 \), was chosen. This grid contained 40 points in the wake, and had the farfield boundary 20 chord lengths away from the surface. Furthermore, the influence of the streamwise grid density variation on the shock location was studied by using a refined, C-type, 337 \( \times \) 91 point grid.

3.1. Steady-state computations

Steady-state solutions at fixed angles of incidence were computed first. The purpose of these computations was not to validate the flow code but to investigate the effect of modeling turbulent and transitional flow effects. In addition, a grid sensitivity study was also performed. Another objective of these solutions was to apply corrections for the presence of the wind-tunnel walls which are absent in the computations. At the outer boundaries of
the computational domain farfield boundary conditions are applied. The effect of the wind-tunnel walls is taken into account by the corrections which were applied by progressively varying both the free stream speed and the angle of incidence until the computed surface pressure distribution matched the experimentally measured values as closely as possible. This preliminary analysis for the steady-state flow yields the modified initial flow conditions and the most appropriate physical models which will be subsequently used to obtain aeroelastic solutions.

### 3.1.1. Viscous flow effects

Limit cycle is an inherently transonic phenomenon caused by aerodynamic nonlinearities of flows containing strong shocks. For flows at large Reynolds number, at small incidences, and thin airfoil sections where viscous losses are small the inviscid flow equations provide a reasonable aerodynamic model for transonic flutter calculations because they include all aerodynamic nonlinearities that can cause limit cycle behavior. In order to demonstrate that viscous effects are important for flows over the thick NLR 7301 airfoil section, an inviscid flow computations are presented first. The inviscid solution was computed on a C-type, 201 × 41 point grid. In contrast to viscous flow solutions, it was found that corrections of the free-stream Mach number and angle of attack cannot yield agreement between the computed and the measured results. The computed inviscid pressure distribution for the same flow conditions which yielded best agreement with the experiment with viscous computations is shown in figure 2. This indicates that viscous effects are important, as the strength and location of the shock on the suction side are clearly missed by the inviscid solver. A closer agreement with the experimental data was achieved in viscous flow computations obtained with the Baldwin–Lomax turbulence model, and assuming fully turbulent flow (see figure 3). The best agreement was obtained for corrected freestream flow conditions $M_c = 0.753$ and $\alpha_c = -0.08$ degrees.

![Figure 2](image)

**Figure 2.** Comparison of the surface pressure coefficient obtained from an inviscid flow solution; $M_c = 0.753$, $\alpha_c = -0.08^\circ$ corrected freestream boundary conditions. Wind-tunnel experiments by Schewe and Deyhle [20].

### 3.1.2. Transitional flow effects

It was found that the fully turbulent result could be further improved by taking transition into account. Detailed data of the transition onset location were not available from the experiment. Therefore, initially the transition-onset location was computed using Michel’s criterion. Michel’s criterion is based on an empirical formula which is valid for subsonic flows. The transition onset for transonic flow was obtained at an unrealistic location almost 60% chord on the suction side. As a result, the shock location at the suction side was predicted further downstream from the location obtained in the experiment. On the pressure side Michel’s criterion predicted the onset location at 19% chord which slightly improved the pressure distribution on the pressure side. From the experimental data, the transition onset location on the pressure side was estimated to be at approximately 40% chord length. It was found that moving the transition onset location further downstream from the 19% chord location predicted by Michel’s criterion, could improve the steady-state result even more. Computations without transition and with forced transition on the suction side at 3% and on the pressure side at 44% chord length are compared with the experiment in figure 3.

![Figure 3](image)

**Figure 3.** Effect of transition modeling; $M_c = 0.753$, $\alpha_c = -0.08^\circ$ corrected freestream boundary conditions. Wind-tunnel experiments by Schewe and Deyhle [20].

### 3.1.3. Effect of turbulence model

Results obtained with the BB [3] and SA [21] turbulence models showed slightly better agreement with experimental results than the solution computed with the BL [2] turbulence model. The computed pressure distributions with different turbulence models, on the baseline 221 × 91 point grid, are compared with the experimental data in figure 4. In the same figure, computed re-
sults on the refined $337 \times 91$ point grid using the SA turbulence model are plotted. The BB and SA turbulence models provided nearly identical pressure distributions. Compared with the BL solution for fully turbulent flow, the pressure distributions computed by the BB and SA turbulence models on the suction side were slightly worse in the range between 4% and 20% chord length, but the agreement in front of the shock was better. On the pressure side, the pressure distribution was in much better agreement with the experiment between the leading edge and 45% chord length, leading to a stronger shock than predicted by the BL model. From 70% chord length to the trailing edge the BL result was closer to the measurements. The steady-state result computed with the SA turbulence model on the refined grid predicted a slightly higher Mach number at the shock locations, but the overall pressure distribution compared well with the one obtained on the $221 \times 91$ point grid.

3.1.4. Effect of transition location

Application of transition modeling with forced transition locations slightly improved the steady-state results, as shown in Figure 5. Again the BB [3] and the SA [21] turbulence models, yielded almost indistinguishable results. Similar to the experience with the BL [2] model, the use of Michel’s criterion on the suction side led to an unrealistic transition onset location. Therefore, transition onset on the suction side was enforced at 3% chord length. On the pressure side, Michel’s criterion predicted a reasonable onset location at 44% chord length which improved the steady-state result near the trailing edge. Computed solutions with the baseline and the refined grid showed very small differences.

Laminar separation bubbles were not predicted in any computations. Separation was found on the suction side close to the trailing edge for all turbulence models and grids. The location of separation onset was computed at 83% chord length with the BL and the BB model, and at 90% chord length with the SA model. A small separation bubble in the shock region due to shock/boundary-layer interaction was found only for the computation with the BL model on the pressure side.

Steady-state computations could be greatly accelerated by using a local-time-stepping scheme, with no measurable difference, in terms of accuracy, to the results of the time-accurate time-stepping scheme. Typically 3000 time steps were required to converge at a Courant number of 30. All computations with the BB and SA turbulence models were performed time-accurately. A Courant number of 1000 was used and full convergence was achieved after 6000 time steps.

3.2. Flutter computations

The steady-state solution computed with the SA [21] turbulence model, the baseline grid and corrected freestream flow conditions matched the experimentally measured surface pressure distribution reasonably well. Correction of the flow conditions and investigations of the effects of modeling physical processes, such as transition and turbulence, was carried out in order to provide for the aeroelastic computations initial conditions which approximate the experiment.

Unsteady computations were performed using the previously presented steady-state results as starting conditions for the aeroelastic analysis. In the experiments of [14,20], the test case no. 77 was run with a freestream Mach number of $M = 0.768$ (close to the transonic dip). For this case, limit-cycle oscillations in pitch and plunge were found. The experiment was conducted at a total pressure of $p_t = 0.45$ bar and a dynamic pressure of $p_{dyn} = 0.126$ bar. A time-averaged angle of attack of $\alpha = 1.28$ degrees was measured for an angle of attack at wind-off condition of $\alpha_0 = 1.91$ degrees. The experimental wind-off condition $\alpha_0$ is equivalent to the spring-
neutral angle of attack, $\alpha_0$, in the numerical simulation. The same free-stream Mach number and the angle of attack corrections applied for the steady-state computations ($M = 0.753$, $\alpha_c = -0.08$ degrees) were used for the unsteady computations. The spring-neutral angle of attack, $\alpha_0c$, was changed until the calculated time-averaged angle of attack was close to the corrected angle of attack of the steady-state computations ($\alpha_c = -0.08$ degrees). The initial energy needed to disturb the airfoil from its rest or steady-state position was derived from the staticimbalance of the aerodynamic moment and the moment of the spring. The nondimensional structural parameters of the experiment used for the unsteady computations are summarized in Table I.

The first series of flutter computations was performed assuming fully turbulent flow. The starting solutions for each turbulence model were the same as given in Figure 4. It turned out that for all turbulence models the NLR 7301 airfoil began to flutter in two degrees of freedom. Limit-cycle oscillations were predicted for all turbulence models. For example, Figure 6 shows the time history of the pitching amplitude which was obtained from a fully turbulent computation with the SA turbulence model.

All computations, independently of the number of grid points, predicted flutter frequencies approximately 1.7% lower and inter-modal phase angles approximately 2.8% lower than the experiment. The phase angle was estimated from the phase of the fundamental frequencies of pitch and plunge predicted by DFT-analysis of the last 10 cycles. Despite this agreement, and for all turbulence models, the computed pitching and plunging amplitudes of the limit-cycle were off by an order of magnitude or more. Computations with and without structural damping showed that damping had no affect on the flutter frequency and the phase angle but decreased the pitch amplitude by 9% and the plunge amplitude by 7%.

The influence of transition on the flutter behavior was studied in a second series of computations on the baseline grid. For these computations, the SA turbulence model was used because it allowed the largest time-steps. Because of lack of measured data, and the inability of Michel’s criterion to predict transition onset in the presence of the strong pressure gradients encountered in transonic flow, the transition location on the lower side was forced at 3% chord length. On the upper side Michel’s criterion was applied. As a result the transition location was recalculated and varied during flutter. Again limit-cycle two-degree-of-freedom flutter was predicted. In Figure 7, the time history of the pitching amplitude shows that the limit-cycle amplitude was slightly higher than in the fully turbulent computations. The inclusion of transition neither improved the prediction of the phase difference angle between pitch and plunge nor affected the over-prediction of the amplitudes (see Table II).

A study of the grid sensitivity was performed, using the $337 \times 91$ point grid, and compared to the fully turbulent baseline grid computations. Pitching and plunging amplitudes on the refined grid were increased by 5% from those predicted on the baseline grid. The inter-modal phase-angle and the flutter frequency were almost the same. The pitching moment-coefficient and the lift-coefficient hysteresis loops (shown in Figures 8 and 9) demonstrate the limit-cycle flutter in two degrees of freedom. Nonlinear effects were clearly more dominant for the pitching moment-coefficient loop than for the lift-coefficient loop.

An inter-modal phase angle of approximately 172 degrees was predicted (see Figure 10). The pitching

Table I. Structural parameters.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$x_p$</td>
<td>0.2500</td>
</tr>
<tr>
<td>$x_a$</td>
<td>0.0485</td>
</tr>
<tr>
<td>$m$</td>
<td>932.90</td>
</tr>
<tr>
<td>$I_a$</td>
<td>33.460</td>
</tr>
<tr>
<td>$k_a$</td>
<td>0.3348</td>
</tr>
<tr>
<td>$k_h$</td>
<td>0.2560</td>
</tr>
<tr>
<td>$l_a$</td>
<td>0.0041</td>
</tr>
<tr>
<td>$b_h$</td>
<td>0.0073</td>
</tr>
</tbody>
</table>

Figure 6. Limit-cycle flutter obtained with the baseline grid.

Figure 7. Effect of transition modeling on the computed limit-cycle flutter.
Table II. Flutter result.

<table>
<thead>
<tr>
<th>Method</th>
<th>$\bar{\alpha}$ [$^\circ$]</th>
<th>$\hat{\alpha}$ [$^\circ$]</th>
<th>$\hat{h}$ [mm]</th>
<th>$f$ [Hz]</th>
<th>$\Phi$ [$^\circ$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exp.$^a$</td>
<td>1.28</td>
<td>0.18</td>
<td>0.65</td>
<td>32.85</td>
<td>176.7</td>
</tr>
<tr>
<td>SA$^b$</td>
<td>0.07</td>
<td>3.78</td>
<td>11.1</td>
<td>32.30</td>
<td>171.8</td>
</tr>
<tr>
<td>SA$^c$</td>
<td>0.03</td>
<td>3.92</td>
<td>12.0</td>
<td>32.20</td>
<td>172.0</td>
</tr>
<tr>
<td>SA$^d$</td>
<td>0.08</td>
<td>3.98</td>
<td>11.7</td>
<td>32.25</td>
<td>171.6</td>
</tr>
</tbody>
</table>

$^a$ Data of the experiment without wind tunnel corrections.
$^b$ Fully turbulent (221 × 91 point grid).
$^c$ With transition (221 × 91 point grid).
$^d$ Fully turbulent (337 × 91 point grid).

Figure 8. Hysteresis loop of lift coefficient obtained with the refined grid.

Figure 9. Hysteresis loop of pitching moment coefficient obtained with the refined grid.

Figure 10. Limit-cycle flutter obtained with the refined grid.

Figure 11. Pitching moment coefficient obtained with the refined grid.

Figure 12. Lift coefficient obtained with the refined grid.

Moment- and lift-coefficient time-histories are shown in figure 11 and figure 12, respectively. The scaled and shifted time-history of the effective angle of attack is also plotted in figure 11. The approximate value of this angle is $\alpha_{eff} = \alpha - \arctan(\hat{h}/U_\infty) - \arctan(x_p\hat{\alpha}/U_\infty)$. The effective angle and the geometric angle are almost coincident because the plunge and pitch contributions are five times smaller than the geometric angle of attack.
and, additionally, almost cancel each other. While the time history of the lift coefficient is almost smooth and harmonic, the time variation of the pitching moment coefficient clearly shows the nonlinear response of the flow field to the motion of the airfoil which causes the limit-cycle flutter. It can be seen that a higher harmonic influence on the pitching moment-coefficient time-history is induced at the minimum of the pitching moment-coefficient distribution which in turn occurs when the airfoil is pitching up and plunging down. This part of the motion corresponds to the portion of the cycle where the magnitude of the effective angle of attack is the highest.

It was shown by Weber et al. [24] that during this period the shock on the suction side becomes stronger and moves upstream causing a shock induced separation. After reversal of the pitch motion, a smooth pitching moment variation is predicted up to the highest pitching moment coefficient. Once the highest pitching moment coefficient is reached, the smooth variation is continued until the pitching motion reaches the lower reversal point. During that time, the airfoil is pitching down and plunging up, corresponding to a low effective angle of attack. Near the reversal point of the pitch and plunge motion, a higher harmonic influence can be seen, corresponding again to the highest magnitude of the effective angle of attack. Looking at one cycle, the dominating nonlinear flow effects occur, as expected, when the airfoil plunging and pitching speed is decelerating, and the magnitude of the effective angle of attack is highest. As an example of the Mach number distribution in the flow field a snapshot at a time when the shock on the suction side causes a large separation of the boundary layer is shown in figure 13.

It is expected that the wind-tunnel walls have a strong influence on the development of the unsteady, transonic flowfield due to their proximity to the airfoil surface. However, the oscillation frequency depends on the natural frequency of the system. Thus, the results of the flutter computation without wall effects but with free stream corrections can be regarded as a first order approximation of the experiment of [14,20]. The computed results are summarized in the following tables. The freestream Mach number, angle of attack, and spring-neutral angle of attack are given in table III. For comparison, the uncorrected values of the experiment are given in the first row of the table. The corrected values, taking into account wind-tunnel effects, are given in the following rows. Flutter-frequency, phase, amplitudes \( \hat{\alpha} \), \( \hat{h} \), and mean angle of attack \( \bar{\alpha} \) are presented in table II.

All the unsteady computations showed that even if the computed time-averaged angle of attack differed from the steady-state angle of attack by 0.1 degree, it had no significant influence on the overall flutter behavior. The unsteady computations were performed on SGI Octane 250 MHz, R10000 workstations and Pentium II-400 Linux PCs. For each cycle of the airfoil motion 2000 to 3000 physical time steps were used, including two Newton subiterations for every time step.

4. Conclusions and outlook

A numerical investigation of the transonic steady-state aerodynamic and flutter characteristics of the NLR7301 supercritical airfoil was carried out. It was found that good agreement with the measured time-averaged pressure distribution could be obtained after correcting the free-stream Mach number and the angle of attack to account for wind-tunnel interference effects. A careful study of the physical parameters influencing the accuracy of the numerical solution of the steady and unsteady flowfield was conducted. It was found that transition can play an important role.

The transonic, two-degree-of-freedom, bending/torsion flutter aeroelastic analysis of the NLR 7301 supercritical airfoil section predicted limit-cycle flutter. The computed limit-cycle amplitudes were caused by the nonlinear features of the flow field, such as shocks and flow separations, and not by the structural model which was linear by definition. The phase angle between pitch and...
plunge and the flutter frequency matched the experimental values quite well, but the computed flutter amplitude exceeded the measured amplitude by an order of magnitude, independent of the number of grid points. Grid independent unsteady solutions were obtained. The discrepancy between the measured and the computed amplitudes are likely due to the following causes which need to be further investigated:

a. omission of unsteady wind-tunnel interference effects and incorrectly chosen corrected free-stream Mach number;

b. incorrectly chosen spring-neutral angle of attack;

c. inaccuracies in modeling complex physical processes, such as flow transition.

Recently, Castro et al. [6] attempted to account for the presence of the wind-tunnel walls and found an improved prediction of the limit-cycle flutter amplitudes of the NLR7301 airfoil. However, further refinement of the porous wall boundary conditions is required before definitive conclusions can be reached.

Acknowledgements

The first author gratefully acknowledges the support of the Deutsche Forschungsgemeinschaft (DFG) and the Naval Postgraduate School. We are indebted to Dr. G. Schewe of DLR Göttingen, Germany, who conducted and analyzed the experiment, for kindly making available as yet unpublished experimental data. Thanks to Dr. A. Knipfer of DLR Göttingen, Germany, for helpful discussions. NAVO provided computing time on the DoD High Performance Computing Systems.

References


